DOI: 10.13009/EUCASS2017-194

Numerical Research of a Propeller Plane Based on Actuator Disc Model

Yifei Zhang, Haixin Chen*, and Yufei Zhang**
**School of Aerospace Engineering, Tsinghua University*
Haidian District, Beijing, 100084, China

Abstract
The flow of a propeller is complex and has obvious influence on the aircraft. Though computer is becoming powerful, numerical simulation on propeller slipstream is still difficult. When doing optimization, actuator disc model could be used to save computation costs and computation time. This paper studies a propeller, then use the actuator disc model to study the slipstream of a propeller plane.

1. Introduction

Nowadays, in the world of air transportation, there is increasing emphasis on Environmentally Responsible Aviation that address more energy efficient and environmentally sustainable with low emissions. In the aerodynamic research of propeller aircraft, the slipstream of propeller and the propeller-plane-interaction are two of the most important and difficult key points. It is becoming increasingly difficult to ignore how the slipstream affect the aircraft aerodynamically during the design of a propeller plane. As a result of unsteady flow and asymmetric flow, the interaction between slipstream and wing, tail and fuselage is very complex [1], causing difficulty in numerical research of propeller planes.

In order to study the propeller slipstream accurately by Computational Fluid Dynamics(CFD) methods, numerous studies have been carried out and researchers have established a series of models to calculate the propeller slipstream. In the early years, momentum disc theory was developed for marine propellers research [2][3][4][5][6][7][8]. Then Blade Element Theory [9][10][11] and Vortex Lattice Method [12][13][14] were developed, to simulate the rotating slipstream in a more accurate way. In the recent years, engineers were using quasi steady method to study propellers [15][16][17]. With the rapid development of computer science, it is becoming more and more possible to use unsteady CFD method to calculate the slipstream flow field, such as Large Eddy Simulation [18][19]. However, it we take massive calculation or engineering design optimization into account, the method of actuator disc can save lots of computation costs, and ensure the accuracy of slipstream relatively [20]. So the actuator disc model has high engineering practicability.

To save computation costs, this paper uses the actuator disc model to carry on the research. The actuator disc model used in this paper is developed from momentum disc theory. Firstly we caculate a real propller, collecte the real flow field data, then we fit the circumferential velocity distribution into actuator disc model, thus the effect of propeller slipstream could be accurately reflected. In this paper, we compare one propeller aircraft with two different states, one with power and one without. The actuator disc method will not enlarge the computation costs. Usually, the rotation direction of the propeller on both sides of a twin-propeller aircraft is the same, thus causing the flow field become asymmetric. Compare the powered results and unpowered results, we can get a conclusion that the propeller slipstream does have great effect on the aircraft.

2. Solver

In this paper, self-developed software NSAWET is used for the CFD simulation. NSAWET solves the Reynolds Averaged Navier-Stokes Equations(RANS) by using lattice finite volume method. NSAWET is a parallel solver based on structural meshes, and can be accelerated by mutigrid method. The turbulent model used is Shear Stress Transport Model. The time march is LUSGS, the numerical scheme is ROE, and the reconstruction is 3-rd MUSCL.

DOI: 10.13009/EUCASS2017-194

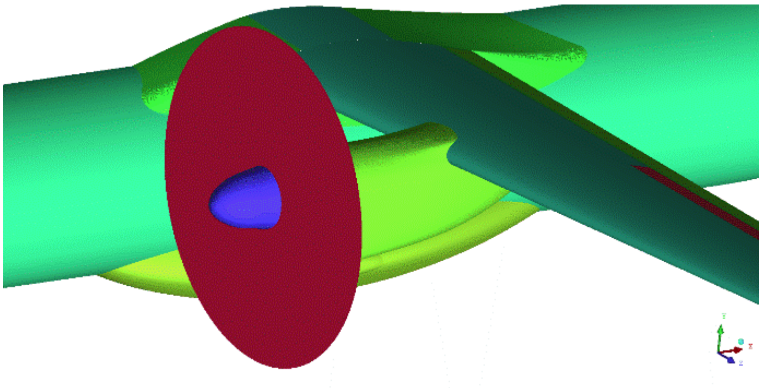

Figure 1: Geometry model of a twin-propeller aircraft

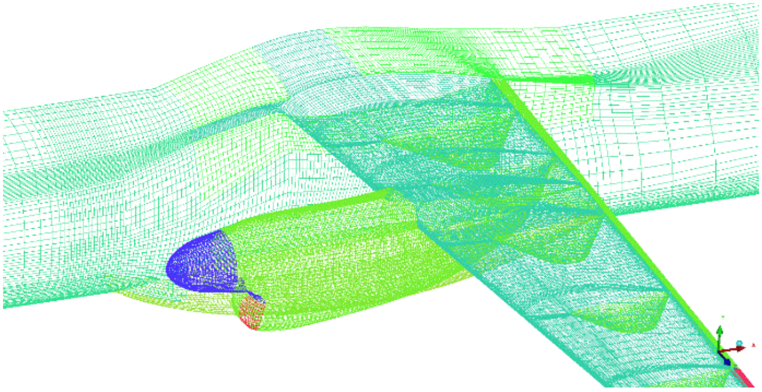

Figure 2: Mesh with no actuator disc, for powerless calculation

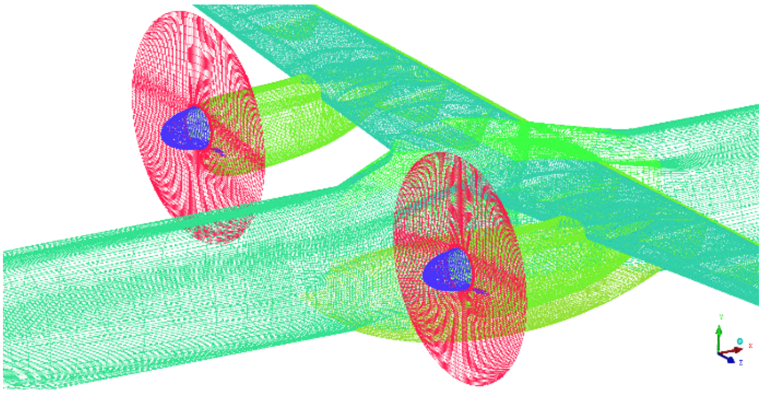

Figure 3: Mesh with actuator disc, for powered calculation

3. Problem Description

This paper focuses on a twin-propeller aircraft, as shown in figure.1. The half model mesh is shown as figure.2, and mesh number is 14.91 million. However, the rotational direction of propellers on both sides is exactly the same, causing asymmetrical slipstream flow field, as a result we will have to calculate the full model(Fig.3) in powered cases. Propeller slipstream is a very complex flow phenomenon, to simplify the slipstream calculation, actuator disc model is used to replace the real rotational propeller. Actuator disc model is similar to boundary conditions, and is used to joint the flow field before and after the actuator disc. To eliminate the effects of meshes, same set of mesh is used on both powerless and powered cases. When calculating powerless cases, the propellers have no boundary conditions. When calculating powered cases, the propellers are actuator disc boundary condition.

DOI: 10.13009/EUCASS2017-194

4. Actuator Disc Model Theory

The first actuator disc method study was reported by W.J.Rankine and R.E.Froude in 1865. They studied a marine propeller and proposed the momentum disc model. When fluid flows pass a propeller, there will be momentum change and energy change. As momentum disc model describes, a propeller can be equivalent to a disc with an infinite number of blades. And the thrust on the momentum disc is evenly distributed when the propeller rotates. In fact, the propeller plane studied in this paper flies at a relatively low Mach number, so that we can ignore the air compressibility and consider air as incompressible fluid. According to momentum disc theory, there are only pressure change and circumferential velocity change across the actuator disc.

According to Bernoulli's principle, far-field inflow and actuator upstream remain energy conservation, and far-field outflow and actuator downstream remain energy conservation:

$$p_\infty+\frac{1}{2}\rho V_\infty^2=p+\frac{1}{2}\rho(V_\infty+V_a)^2 \tag{1}$$

$$p+\Delta p+\frac{1}{2}\rho(V_\infty+V_a)^2=p_\infty+\frac{1}{2}\rho(V_\infty+V_b)^2 \tag{2}$$

In Eq.(1) and Eq.(2), p_∞ , ρ , V_∞ are far-field inlet static pressure, air density and inlet velocity ; V_a , V_b are velocity increment at actuator upstream and downstream ; p , Δp are static pressure just before the actuator disc and the pressure change caused by the actuator disc.

According to Eq.(1) and Eq.(2), we can get :

$$\Delta p=\rho V_b(V_\infty+\frac{V_b}{2}) \tag{3}$$

The propeller thrust is contributed by the pressure increment on the actuator disc :

$$T=A\Delta p \tag{4}$$

In Eq.(4), T is propeller thrust, A is the front channel area of actuator disc.

According to energy conservation and Eq.(1)(2)(3)(4), the following can be deduced :

$$V_b=2\cdot V_a \tag{5}$$

In the year 1920, Betz.A added circumferential velocity change into Froude's momentum disc model [22]:

$$\Delta V_\theta=\omega\cdot r \tag{6}$$

In Eq.(6), ΔV_θ is the circumferential velocity change, ω is the propeller's rotational speed, r is the local rotational radius.

The actuator disc model requires the following assumptions:

1) In the actuator disc model, flow is steady;
2) Across the actuator disc, fluid density remains the same;
3) Across the actuator disc, pressure increment exists and can be obtained by thrust;
4) Across the actuator disc, to ensure mass conservation, axial velocity remains the same;
5) Across the actuator disc, to simulate the propeller slipstream, the circumferential velocity changes.

The disc model above[Eq.(1)~(6)] is a rigid-body rotational velocity model, and is widely used on fans and in water [21][22]. However, for an air propeller, since the low air viscosity and low blade solidity, airflow after the blades will not follow them well in rotational direction.

In the circumstance of air propellers, circumferential velocity after blades does not follow $v_\theta=\omega\cdot r$. Actually, the velocity near propeller hub and tip is very low. In order to improve the actuator calculation pricision, blade element theory could be used, which means calculating the lift coefficient and drag coefficicent of airfoils at different blade radius. We can also calculate a individual propeller, then analyze the pressure increment and velocity increment caused by blades, and fit the pressure and velocity increment onto actuator disc model. So that the slipstream could be well simulated. The second method is used in this paper.

DOI: 10.13009/EUCASS2017-194

5. Computation and Discussions

5.1 Actuator Disc Model Computation

Numerical calculations are carried out for an individual propeller. The blade angle is 45.3° , and the CFD model and mesh are shown as figure.4 and figure.5.
The propeller advance ratio is known as $J=V/nD$. $V[m/s]$ is inflow velocity, and $n[r/s]$ is propeller rotational velocity, $D[m]$ is propeller diameter.

$$n=V/JD \quad \textbf{(7)}$$

The propeller thrust coefficient is known as $T_c=T/qS$. $S[m2]$ is aircraft reference area, and $q[Pa]$ is inflow dynamic pressure. So the pressure increment can be deduced:

$$\Delta p=T_c qS/A \quad \textbf{(8)}$$

A real rotating propeller is calculated by steady method. The interface between rotational domain and static domain is mixing plane. After the computation is completed, the downstream interface is selected for data processing. The circumferential velocity distributions of all grid points are counted, and polynomial fitting is done with the local radius $r[m]$ as the independent variable.

$$\Delta V_\theta/V_\infty=A_4(2r/D)^4+A_3(2r/D)^3+A_2(2r/D)^2+2A_1r/D+A_0 \quad \textbf{(9)}$$

The CFD result is shown as figure.5, and circumferential velocity distribution is shown as figure.6.

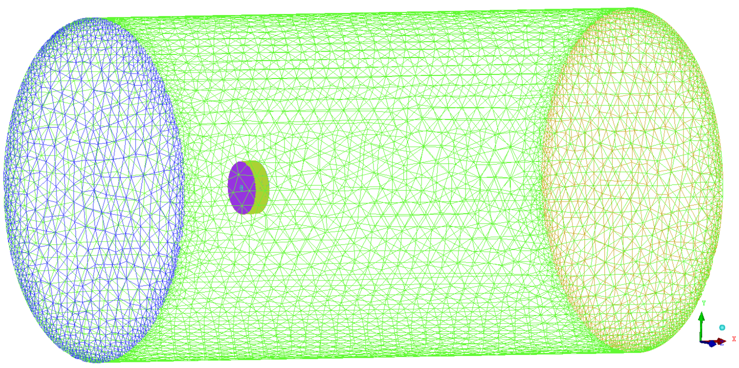

Figure 4: Mesh of an individual propeller solving

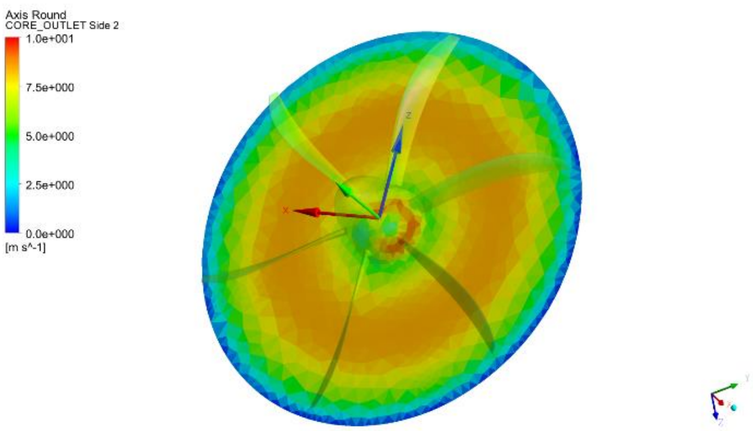

Figure 5: Circumferential velocity contour of the propeller

DOI: 10.13009/EUCASS2017-194

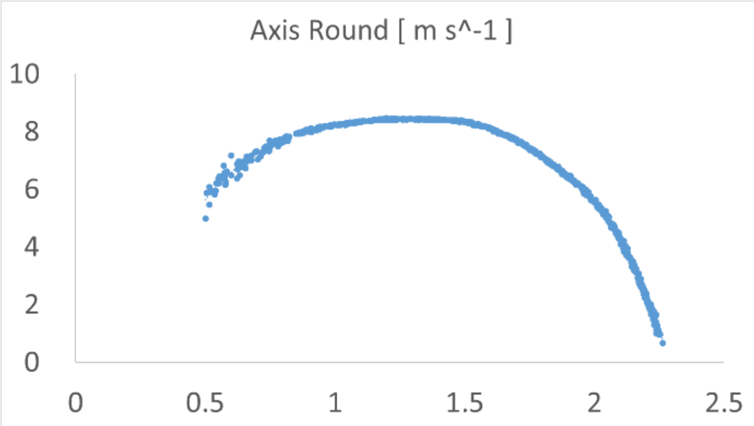

Figure 6: Circumferential velocity distribution for the actuator disc model

Computation Mach number is 0.2 and Reynolds number is 1.9M. Cruising speed is 70m/s and the angle of attack is 0° . The actuator disc model used is as Eq.(8) and Eq.(9) describe in aircraft slipstream simulation, and the results are figure.7 and figure.8.

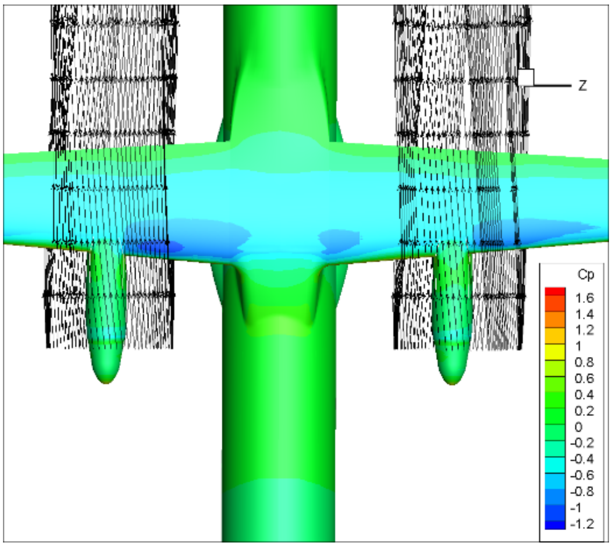

Figure 7: Slipstream and pressure coefficient contour calculated by using actuator disc model, upper surface

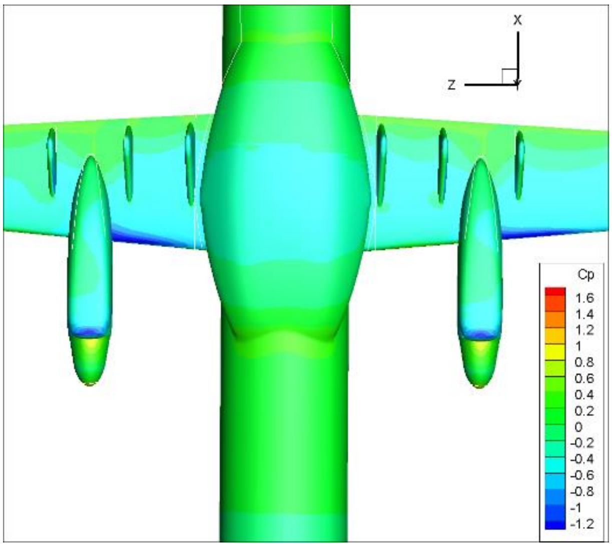

Figure 8: Pressure coefficient contour calculated by using actuator disc model, lower surface

DOI: 10.13009/EUCASS2017-194

As can be seen from figure.7, the slipstream simulated by actuator disc model is acceptable and rotates in anticlockwise direction. At the same time, from figure.7 and figure.8, we can see that under the influence of slipstream, the attack angle near propeller is changed, causing different pressure and asymmetrical flow field. The actuator disc model is used in subsequent calculations.

5.2 Slipstream Analysis

Powerless cases and powered cases have been well calculated, and the angle of attack varies from -8° ~18° . The lift coefficient curve, the drag coefficient curve and the torque coefficient curve are shown in fiture.9, figure.10 and figure.11. Compared to wind tunnel results, the CFD calculations are in good agreement with the experiment.
From the lift coefficient curve and drag coefficient curve, it can be seen that the lift coefficient and drag coefficient of both experiment and calculation are slightly larger when there is power. That is, the propeller slipstream increases the lift on the wing, and brings extra drag to the fuselage, wings and nacelles. However, at circumstance of large angle of attack, the data calculated by CFD and experiments have some deviations. Preliminary analysis shows that, the wind tunnel model will be deformed when the angle of attack is large, as a result the lift coefficient decreases and the drag coefficient increases.

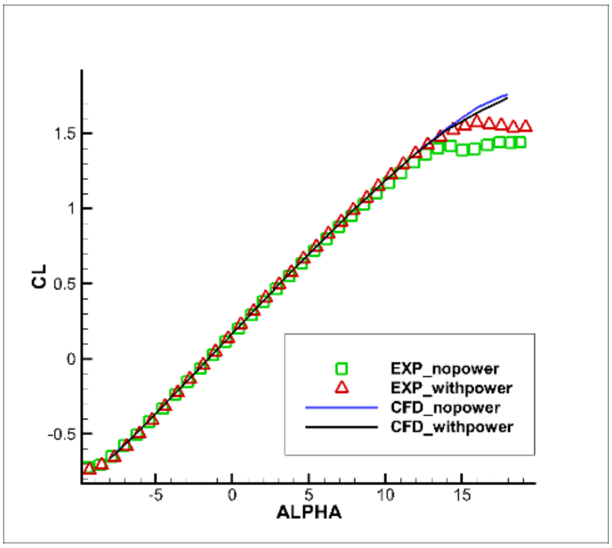

Figure 9: Lift coefficient curve

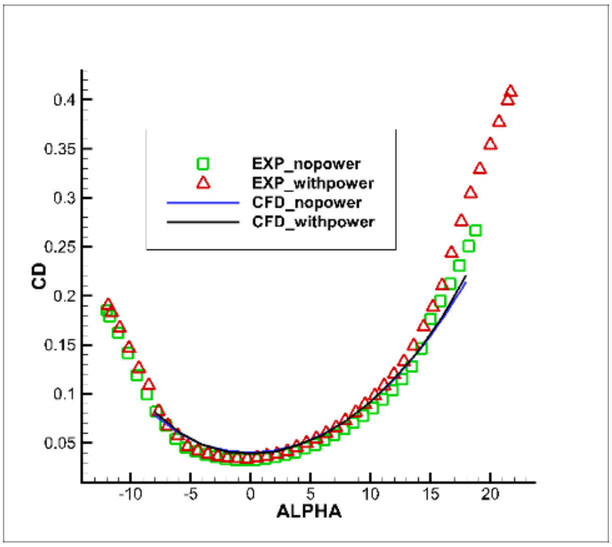

Figure 10: Drag coefficient curve

DOI: 10.13009/EUCASS2017-194

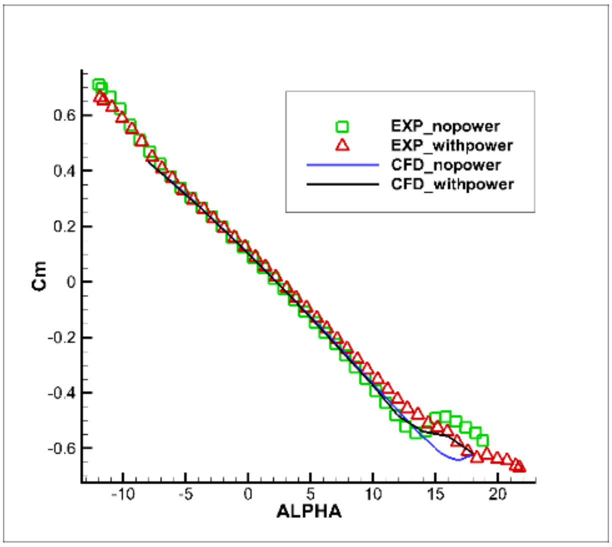

Figure 11: Torque coefficient curve

The friction coefficient contours are shown in figure.12. By comparing contours, for powered case at 6 degrees angle of attack, the flow on upper wing surface begins to separate under the influence of upwash slipstream, but flow will not separate under powerless condition. Meanwhile, at same angle of attack, downwash slipstream helps to reduce the separation area on upper wing surface.

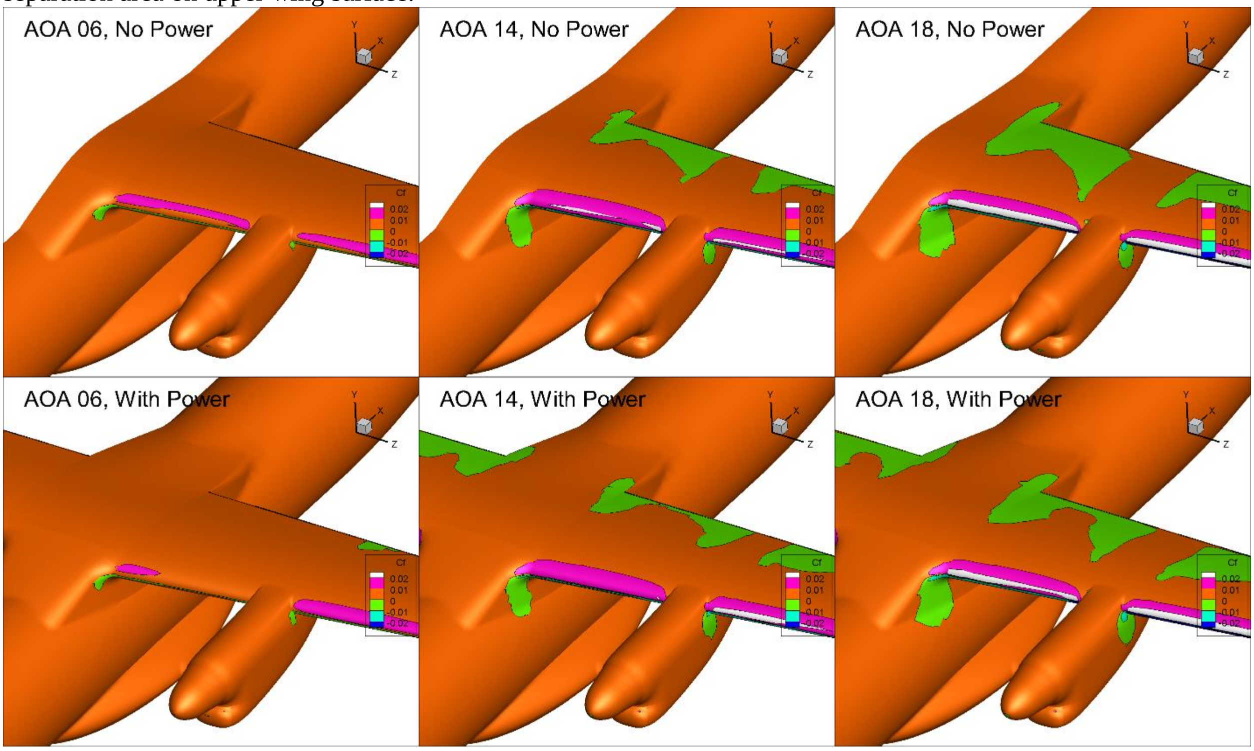

Figure 12: Friction coefficient contour, upper wing surface

Compare the powered calculation results at different angle of attack, large flow separation appears at the right side of horizontal tail, as is shown in figure.13. As the angle of attack increases, the separation area increases, causing obvious asymmetrical flow. This phenomenon is caused by propeller slipstream.

DOI: 10.13009/EUCASS2017-194

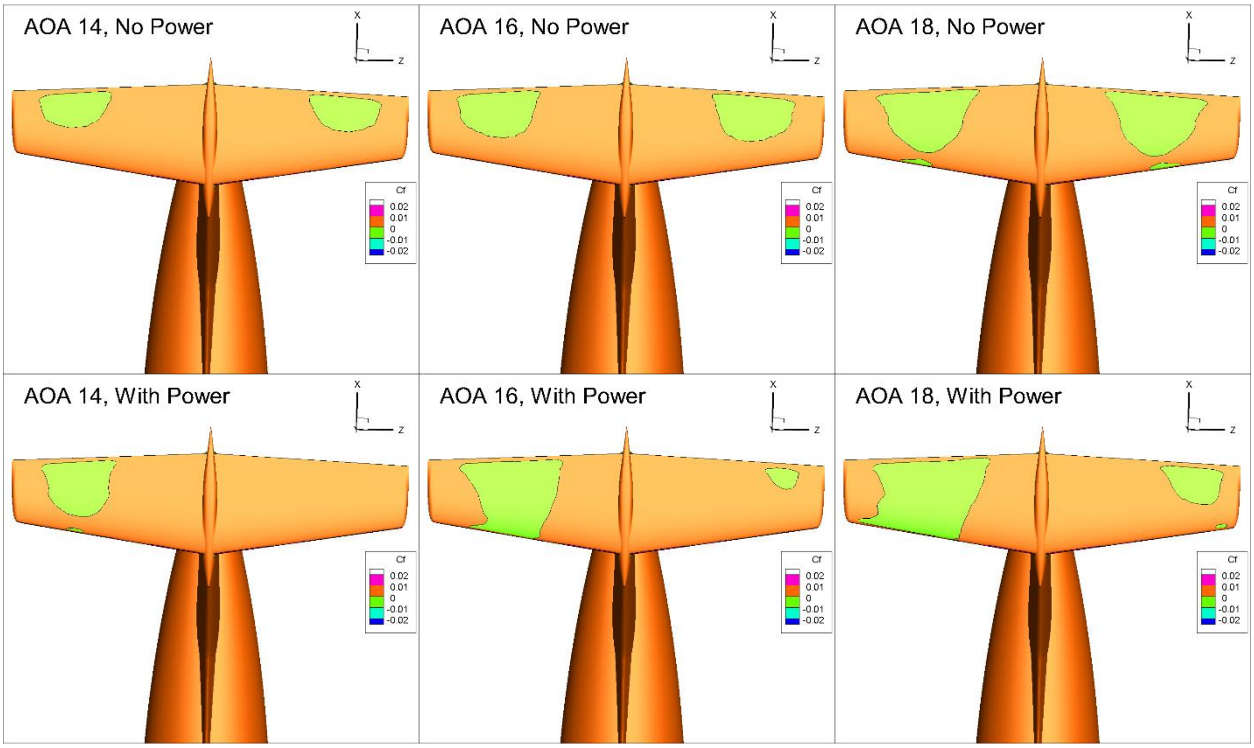

Figure 13: Friction coefficient contour, horizontal tail

6. Conclusions

In this paper, a numerical research is carried out on a twin-propeller aircraft by actuator disc model, meanly focusing on the slipstream influence, and results show that:

1) Relatively accurate CFD results can be obtained by calculating a propeller in advance and then substituting the data into actuator disc model.
2) The propeller slipstream will increase the lift and drag of the aircraft. The flow on upper wing surface tend to separate in advance, which is affected by the upwash slipstream.
3) The propeller slipstream will cause asymmetrical separation on left and right horizontal tail. This asymmetry increases with angle of attack

References

[1] Daniel C.Mikkelson, Glenn A.Mitchell and Lawrence J.Bober. 1984. Summary of Recent NASA Propeller Research. NASA Technical Memorandum 83733. Lewis Research Center.

[2] Giulio Romeo, Giacomo Frulla, Enrico Cestino and Guido Corsino. 2004. HELIPLAT: Design, Aerodynamic, Structural Analysis of Long-Endurance Solar-Powered Stratospheric Platform. *Journal of Aircraft*. Vol.41, No.6: 1505-1520.

[3] H.C.Chen and S.S.Samant. 1985. Flow Simulations Using Euler Equations for Nacelle-Propeller Configurations in a Wind Tunnel Environment. In: *AIAA 18th Fluid Dynamics and Plasmadynamics and Lasers Conference*. AIAA-85-1678.

[4] F.Moens and P.Gardarein. 2001. Numerical Simulation of the Propeller/Wing Interactions for Transport Aircraft. In: *AIAA 19th Applied Aerodynamics Conference*. AIAA-2001-2404.

[5] Xu Jing, Yang Yong and Zuo Suihan. 2008. Numerical Simulation of Propeller/Wing-nacelle Configuration Interaction. *Aeronautical Computing Technique*. Vol.65, No.3: 65-67.

[6] Duan Yiqian and Shi Aiming. 2012. A New and Effective Actuator Disk Model Approach for the Simulation of Propeller Slipstream. *Journal of Northwestern Polytechnical University*. Vol.30, No.6: 841-846.

DOI: 10.13009/EUCASS2017-194

[7] Zuo Suihan and Yang Yong. 2007. Numerical Simulation of Propeller/High-lift System Interaction. *Aeronautical Computing Technique*. Vol.37, No.1: 54-57.

[8] Bryan Farrar and Ramesh Agarwal. 2014. CFD Analysis of Open Rotor Engines Using an Actuator Disk Model. In: *AIAA 52nd Aerospace Sciences Meeting*. AIAA-2014-0408.

[9] Lu Hao. 2014. Numerical Investigation of Propeller Slipstream Effects with Actuator Disk Model. Master Thesis. Nanjing University of Aeronautics and Astronautics, College of Energy and Power.

[10] Li Bo, Liang Dewang and Huang Guoping. 2008. Propeller Slipstream Effects on Aerodynamic Performance of Turbo-prop Airplane Based on Equivalent Actuator Disk Model. *ACTA Aeronautica et Astronautica Sinica*. Vol.29, No.4: 845-852.

[11] Xia Zhenfeng, Luo Song and Yang Yong. 2012. Numerical Simulations of Propeller Slipstream Flows using Actuator Disk Theory. *ACTA Aerodynamica Sinica*. Vol.30, No.2: 219-222.

[12] Yang Guowei and He Zhidai. 1995. Flow-Field Calculation of Propeller Slipstream About Vortex Contraction. *ACTA Aerodynamica Sinica*. Vol.13, No.1: 83-86.

[13] E Qin, Yang Guowei, Li Fengdai and He Zhidai. 1996. Numerical Calculation of Interaction of Propeller Slipstream on Flowfield over a Complete Aircraft. *ACTA Aeronautica and Astronautica Sinica*. Vol.17, No.4: 439-442.

[14] E Qin, Yang Guowei, Li Fengdai and Fu Dawei. 1997. On Coupling Effect of Two Vortex Systems of Chinese Aircraft with Turbo-Propellers. *Journal of Northwestern Polytechnical University*. Vol.15, No.4: 511-516.

[15] Xu Jiakuan, Bai Junqiang, Huang Jiangtao, Qiao Lei, Dong Jianhong and Lei Wutao. 2014. Aerodynamic Optimization Design of Wing Under the Interaction of Propeller Slipstream. *Acta Aeronautica et Astronautica Sinica*. Vol.35, No.11: 2910-2920.

[16] J.-M. Bousquet and P.Gardarein. 2003. Improvements on Computations of High Speed Propeller Unsteady Aerodynamics. *Aerospace Science and Technology*. 7: 465-472.

[17] Arne W.Sturmer. 2006. Unsteady CFD Simulations of Propeller Installation Effects. In: *42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*. AIAA-2006-4969.

[18] Qiao Yuhang, Ma Dongli and Li Zhi. 2015. Unsteady Numerical Simulation of Propeller/Wing Interaction. *Journal of Aerospace Power*. Vol.30, No.6: 1366-1373.

[19] W.Shawn Westmoreland, Robert W.Tramel and Jennie Barber. 2008. Modeling Propeller Flow-Fields Using CFD. In: *46th AIAA Aerospace Sciences Meeting and Exhibit*. AIAA-2008-402.

[20] Feng Jiang. 2003. Computational Analyses for an Advanced Propeller Powered Theater Transport. In: *21st Applied Aerodynamics Conference*. AIAA-2003-4081.

[21] Hu Xuan and Zhang Qiang. 2015. Aerodynamic Excited Forces and Dynamic Characteristics of Eccentric Turbine Rotor in Non-uniform Inlet Flow Conditions. *Science Technology and Engineering*. Vol.15, No.7: 115-121.

[22] Betz.A. 1920. Development of the Inflow Theory of the Propeller. *Kinetic Theory of Gases*: McGraw-Hill Book Co..